

Student CNC Guide



KTH Royal Institute of Technology
Viktor Stenberg
viktor.stenberg@outlook.com
Stockholm, Sweden, 2015-12-07

Table of Contents

Table of Contents	i
Introduction.....	ii
CNC Machining Safety	1
Technical Glossary	2
What is a CNC Milling Machine?	3
Intro	3
Coordinate Systems	5
Absolute Coordinate System	6
Work Coordinate System	7
G - Code Programming Language	8
Tools	9
Fixtures	12
Machine Overview	14
The CNC Mill	14
Tools and Toolholders.....	19
Accessories	23
Mach 3 Overview.....	25
Intro	25
Jogging.....	30
Setup Work Coordinate System Origin.....	30
MDI Overview.....	31
Wizards Overview	31
Using the Auto Tool Zero Probe	31
Using the Touch Probe.....	32
CAD/CAM Software Overview	34
Intro	34
Autodesk Fusion 360.....	34
Autodesk Fusion 360 Design Workflow.....	35
Workflow of Machining a Part.....	39
Speeds and Feeds Table.....	43
Prefix Reference	45
G - Code Reference	46
M - Code Reference	47

Introduction

This manual aims to quickly make you successful in operating the Student CNC Mill and making very precise and high quality parts for your project.

The first sections of the manual takes you through the very basics of what a CNC Mill is and what operations it can do, the basics of how the machine works, what G - Code is, the different types of tools that can be used and other practical things you need to know.

This is followed by an overview of the actual Student CNC Mill. Clear diagrams with key components is marked out and explained. The available accessories and tools are reviewed.

Next the control software is explained, how it works and the minimum features needed to know.

This is followed by a quick introduction of the CAD/CAM software. Then the workflow of designing a part in Autodesk Fusion 360 and generate the G - Code for the CNC Mill is presented.

Now that the fundamentals of the CNC process has been explained, a step by step section is presented which shows all of all the steps needed to perform in order to go from drawing to finished part.

A number of useful tables can also be found in the manual.

I hope that the manual enables you to succeed in operating the CNC Mill!

Sincerely,
Viktor Stenberg
Author of Student CNC Guide

CNC Machining Safety

When a CNC machine is operated properly, using the machine is a relative safe process. *This is only under the circumstances that the following rules are followed!*



Safety Rules

- 1. Do not operate the machine if you are not sure of what you are doing, call for help by some who can!**
- 2. Always wear safety glasses when you are near the machine!**
A broken cutter / chip / material can shoot out of the machine during operation.
- 3. Always make sure that the spindle is tightened!**
A loose tool will cause vibrations and can be shoot away during cutting.
- 4. Make sure the stock material is properly tightened!**
The material need to be firmly secured to the table.
- 5. Hit the Emergency Stop / Escape Key if something goes wrong!**
This quickly stops the machine preventing danger.
- 6. Never put your Fingers / Hand / Head near the spinning cutter!**
Don't risk your fingers, stop the machine by Feedhold, adjust then resume.
- 7. Never leave the machine unattended!**
If something goes wrong, you need to be around to stop the machine.

Technical Glossary

Stock Material	The raw material the part should be machined out of.
G - Code	The programming language of the CNC Mill.
Feedrate	The speed of the axis.
CAD	Software to draw parts in 3D.
CAM	Software to generate G - Code from the CAD files.
MDI	Manual G -Code input in controlling software (Mach 3).
DRO	Digital Read Out. Displays a number, typically an axis coordinates value.

What is a CNC Milling Machine?

Intro

A CNC Milling machine is a computer controlled machine that can be used to make very precise parts.

The machine works by *removing* material from the *workpiece* with a *rotating cutting tool*.

The machine does this by guiding the tool in all three directions of the *cartesian coordinate system*, that meaning along the X, Y and Z axis.

The arrows in the figure below illustrate the three directions the CNC Mill can move in.

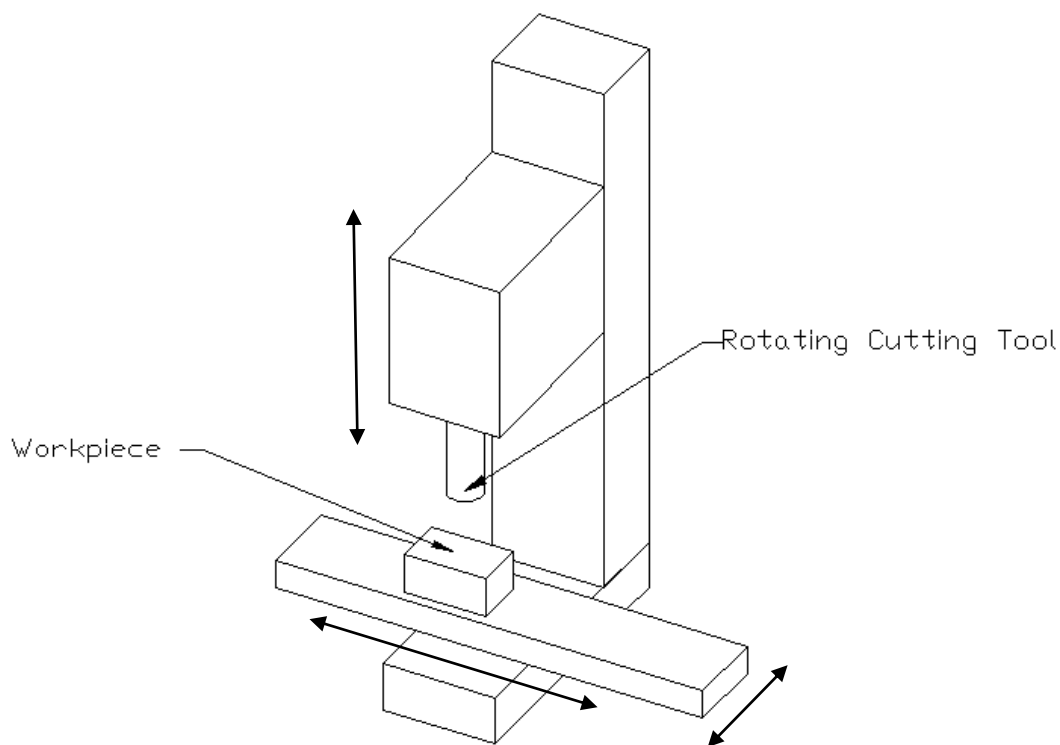


Figure 1: Illustration of a CNC Mill

The CNC Mill can machine many different materials, examples are steel, aluminum, brass, copper and plastic. A Mill is *not designed* for cutting wood and should not be used for that.

Examples of operations that can be made in a CNC Mill is to cut a profile shape, engrave text, mill a 3D surface, drill holes and mill bearing pockets.



Fig 2: Example of CNC Milling

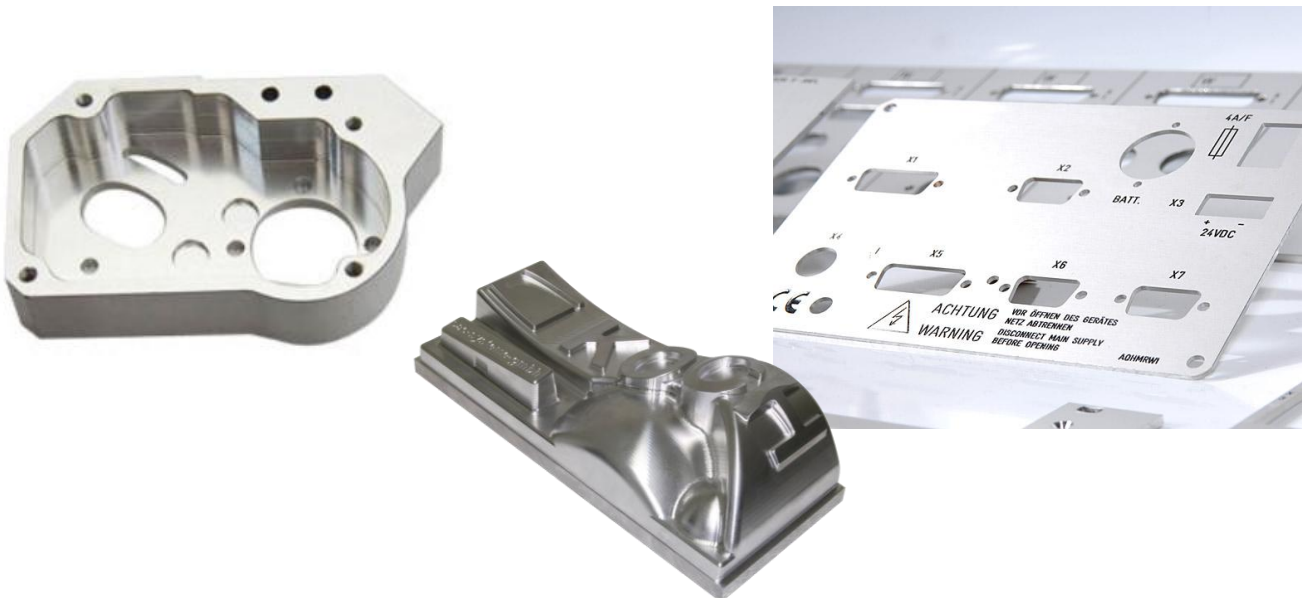


Fig 3: Example of CNC Milled parts

Coordinate Systems

The CNC Mill can as previously mentioned move in the X, Y and Z directions of the cartesian coordinate system. The coordinate system is defined as illustrated below.

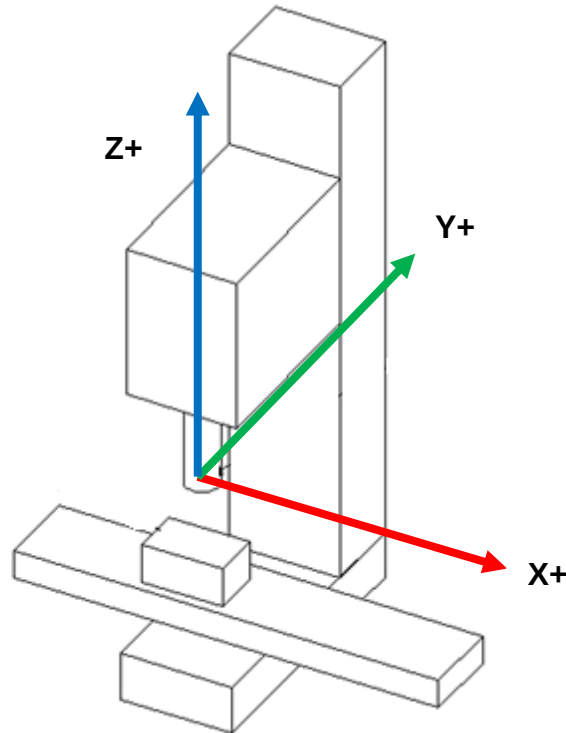


Figure 4: Coordinate system

Note that the coordinate system is defined *relative to the tool motion!*

This means that when the machine for example is cutting in the positive X direction, the tool should move to the right which is the positive direction as defined by the coordinate icon in Fig 4.

Since the tool cannot move in any other direction then up and down due to the design of the mill, the table has to move.

The table and the stock attached to it will thus have to travel in the opposite direction, i.e. to the left when the mill is cutting in the positive X direction in order for the relative motion to be to the right. The same logic applies to the Y axis, but not the Z axis.

Absolute Coordinate System

The absolute coordinate system of the CNC Mill is called the *Machine Coordinate System*, and its *Origin* is located at the endpoint of the three axis's. From the machine origin the X and Y axis can only move in the positive direction and the Z axis can only move in the negative direction.

The origin of the machine coordinate system is called *Home Position*. Every time the machine starts up the operator should perform a homing cycle to find the Home Position. This enables the machine to know where it is. The machine finds the Home Position by slowly driving each axis to their extreme limit (-X, -Y, +Z). As each axis reaches its mechanical limit, a sensor triggers and stops the motion. When this has been done to all three axes, the position of the machine is at the origin of the machine coordinate system.

It is very important that the machine is referenced to its absolute position since the control computer and G - Code programs base some movements with respect to this coordinate system. Safe axis limits are also based on this coordinate system.

The operator must perform a Homing Cycle on every startup of the machine!

Figure 5 illustrates the origin / home position of the machine coordinate system.

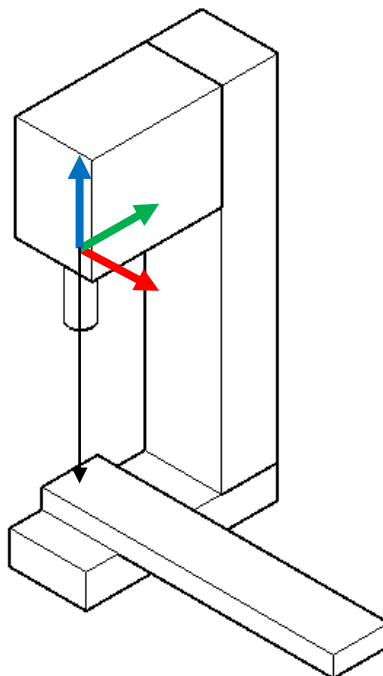


Fig 5: Machine Coordinates origin

Work Coordinate System

When programming the CNC Mill it is useful to define a custom coordinate system that can be set anywhere on the part to be milled.

A Work Coordinate System can be setup by moving the machine so that the cutter is in the appropriate place and call this the origin of the Work Coordinate System

The origin of the Work Coordinate System will typically be set at the center of the part and with the tip of the cutter directly above the part.

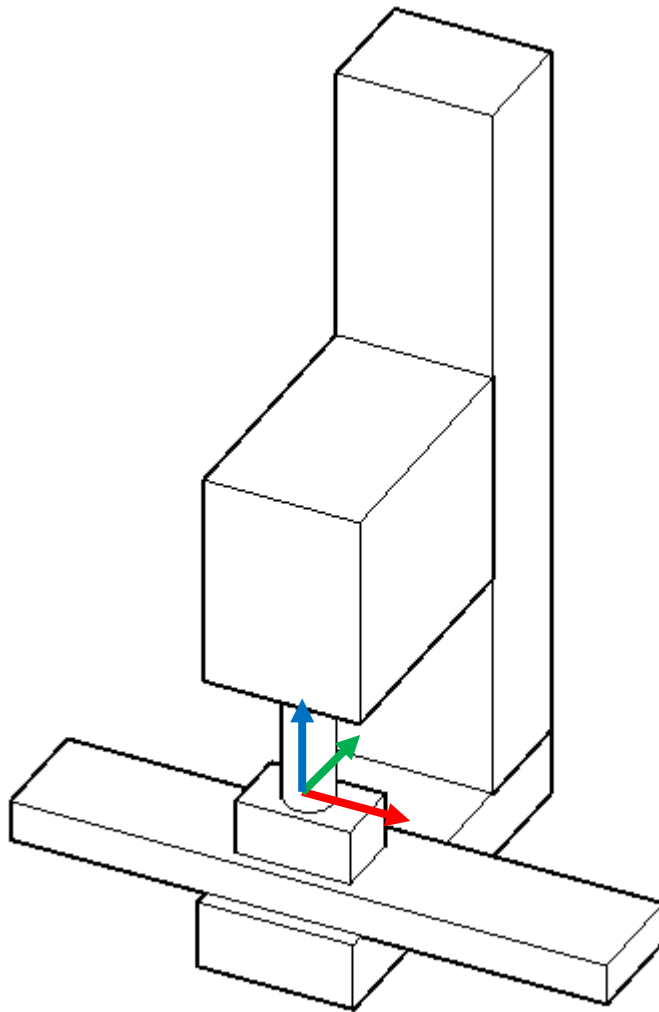


Figure 6: Work Coordinate System

G - Code Programming Language

So how do you make the machine move and actually machine a part? This is done by running a CNC program. A CNC program is loaded into the Control Computer, which then is executed causing the machine to perform the movements programmed in the file.

The *G - Code Programming Language* is the name of the language that the CNC machine can understand and convert into actual motion of the machine. The Control Computer reads the G - Code file line by line and commands the machine to do the movements.

The language consists of a set of codes called G - Codes and M - Codes, each instruction have a specific function, for example to command a motion from one point to another.

The following is an example of a G - Code program:

; Example program

G54 ; Use G54 work coordinates offset

G90 ; Absolute positioning

G0 X0 Y0 Z0 ; Rapid move to (0,0,0)

G1 Z-1 F50 ; Lower the tool to Z = -1 at a speed of 50 mm/min

G1 X30 Y30 F100 ; Move to X = Y = 30 mm simultaneously, 100 mm/min

M30 ; End of program

The G - Code program can be written manually in a file that then is executed, or inputted line by line and executed in real time through the *MDI* input of the Control Computer. These methods are useful if some very simple operations are to be made.

It's much more common to use a *CAM Software* to generate the G - Code. The CAM software imports a 2D or 3D model and generates a G - Code file to cut the part. There is also a simpler type of CAM software called *Wizards* that will generate G - Code for common operations like milling a circular pocket or drilling a hole. Wizards are very handy if a relatively simple part with few operations are to be made. These can be found in the CNC – Software and do not need a CAD model.

A complete list of all the programming codes can be found under the sections 'G - Code Reference', 'M - Code Reference' and 'Prefix Reference'.

Tools

The CNC machine uses different tools to perform various operations on the part. The following list describes some common tools.

Square Endmill	The most common tool. Used to make square cuts.	
Ball Endmill	Used to do 3D milling.	
Bull Nose Endmill	Produces a small chamfer at the bottom of the cut.	
Twist Drill	Used to drill holes.	
Center Drill	Used to make a small guide hole for a Twist Drill. This makes sure the Twist Drill does not wobble.	
Face Mill	Used to face mill, face milling produces a flat surface on the part.	
Engraving Tool	Used to engrave text on parts or engrave circuit boards (PCBs). Also called V-bit.	
Chamfer Tool	Used to produce a chamfer on a hole or around the profile of a part.	

An endmill can be Center Cutting or Non Center Cutting. A center cutting endmill can be plunged straight down into the material since its cutting edges / flutes go all the way into the center. A non center cutting endmill cannot be plunged and must be ramped into the material.

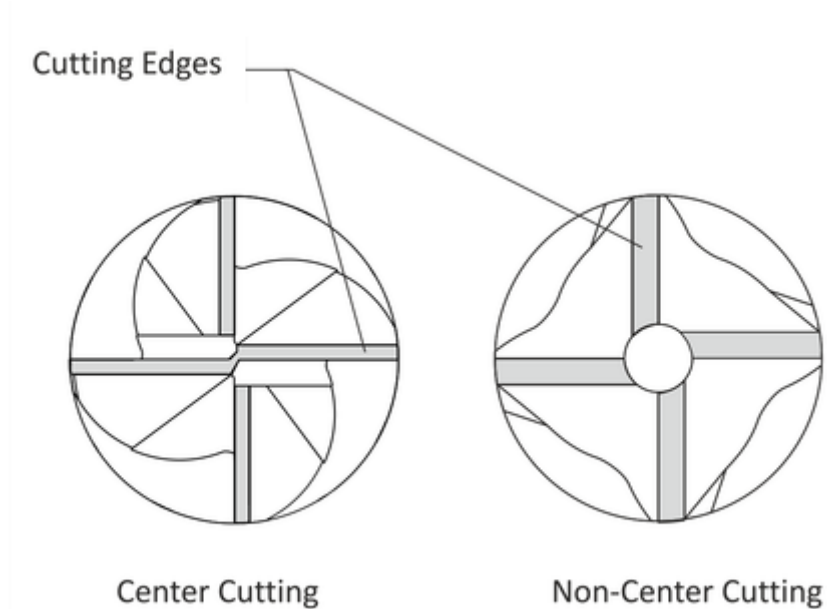


Fig 7: Center Cutting vs Non Center Cutting

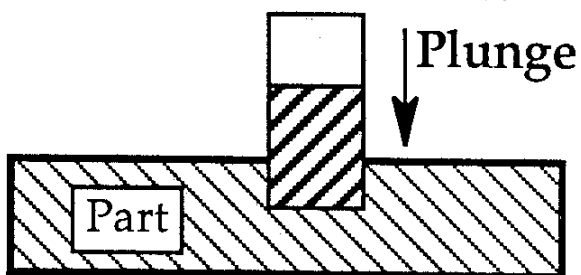


Fig 8: Endmill plunging into material

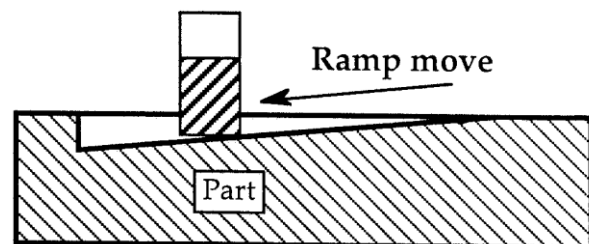


Fig 9: Endmill ramping into material

Climb milling is preferred to use on a CNC milling machine.



Fig 10: Climb Milling vs Conventional Milling

The number of cutting flutes / edges on the endmill determines the type of material it is designed to cut.

No. of Flutes vs Intended Material to Cut	
2 Flutes	Aluminum
3 Flutes	Plastic
4 Flutes	Steel

Fixtures

The stock material should be clamped to the table in one of the following ways.

Never clamp the stock material directly on the table! Always put a safety sheet of material under the stock in order to prevent the mill cutting into the table!

Clamping on Table

The stock can be clamped onto the table with clamps.

Always place a safety sheet of material under the stock so that the mill does not cut accidentally into the table!

This also enables to cut a bit deeper than the thickness of the stock.

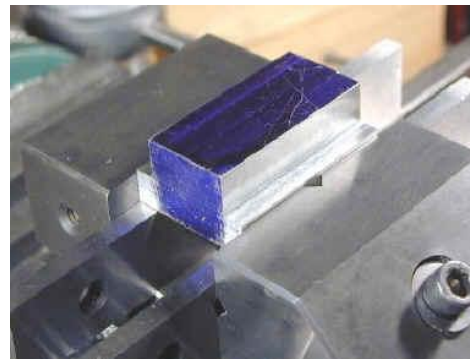


Clamping in Vise

The stock material is often clamped between the jaws in the vise.

Use a pair of parallels to make sure that the stock is clamped parallel to the table.

Parallels are a pair of flat bars that can be positioned at the bottom of the vise for the stock material to sit on top of.



Clamping in Vise with Soft Jaws

A pair of soft jaws can be made from aluminum and installed in the vise. The soft jaws can then be machined to accept a stock material with non-square shape.

This should not be done to the standard steel jaws of the chuck!



Clamping in Chuck

A lathe chuck can be clamped to the table in which the stock material can be clamped in.

This is useful to do lathe like machining operations on round stock.



Machine Overview

The CNC Mill

The Student CNC machine is a SIEG KX3 CNC Mill with the following specifications.

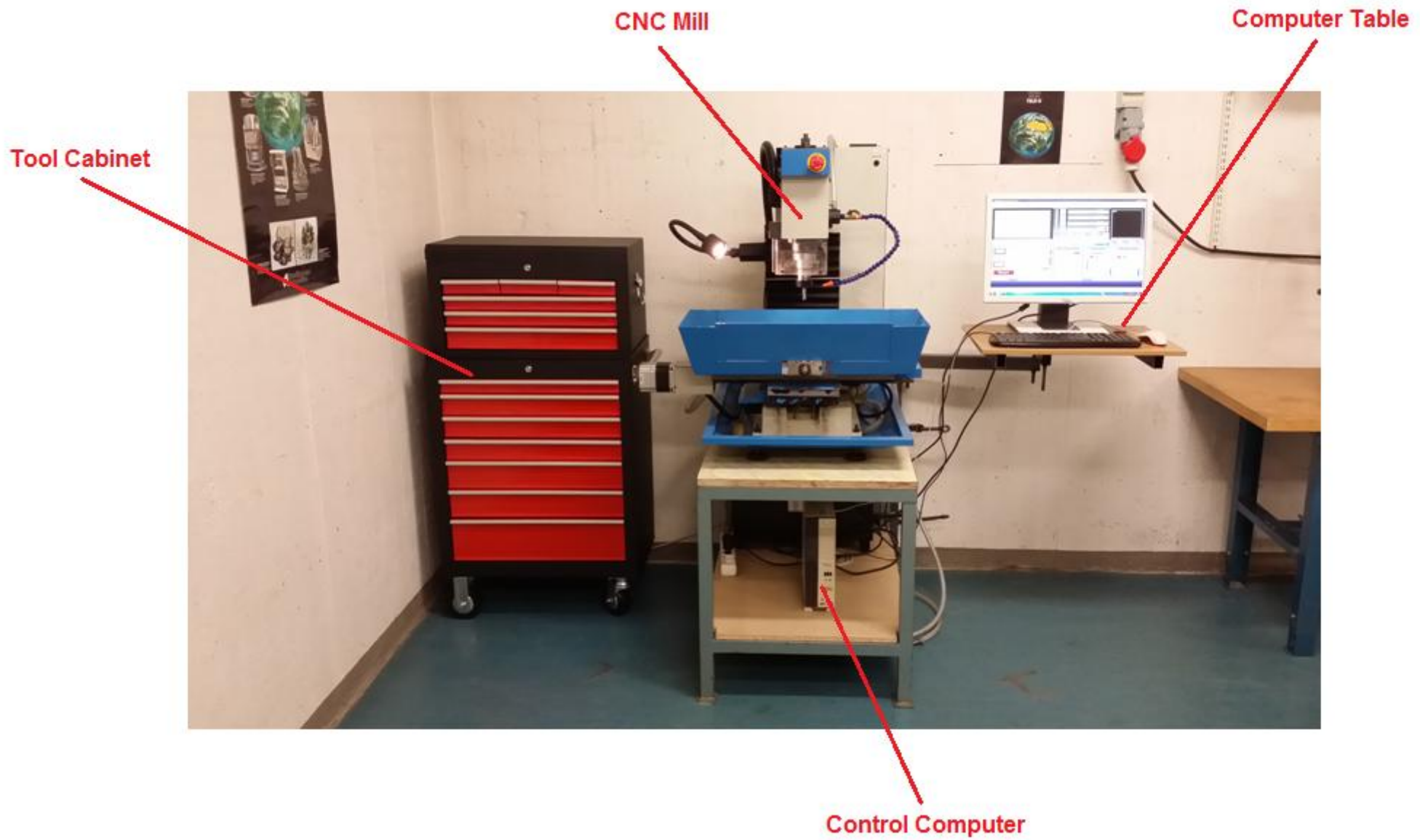
X Travel:	290 mm
Y Travel:	125 mm
Z Travel:	275 mm
Maximum Feedrate:	2000 mm/min
Spindle Motor Power:	1000 W
Spindle Speed:	100 – 5000 RPM
Spindle Taper:	R8
End Milling Capacity:	25 mm
Face Milling Capacity:	80 mm
Drill Capacity:	20 mm

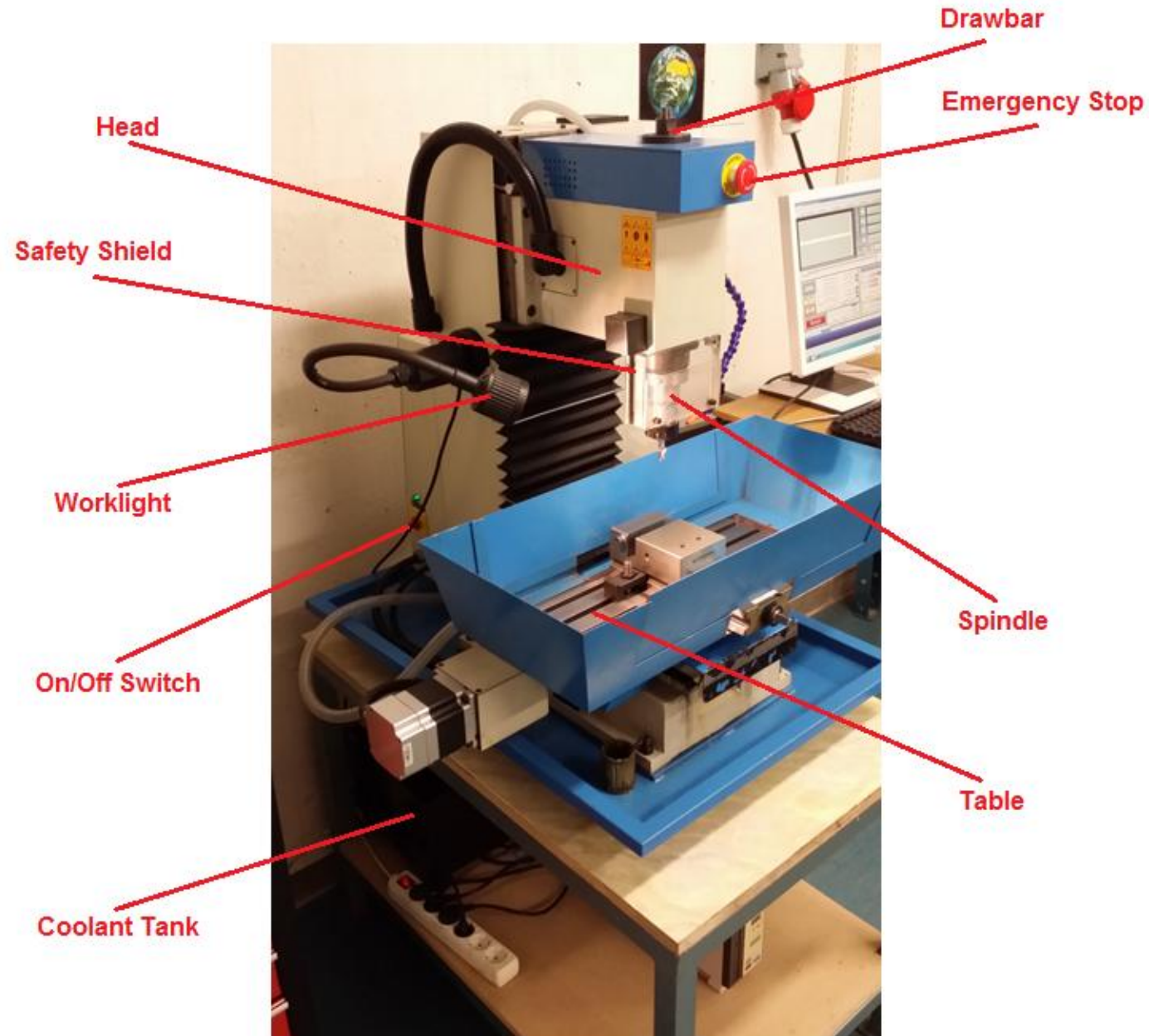
Maximum size of stock material that can be machined:

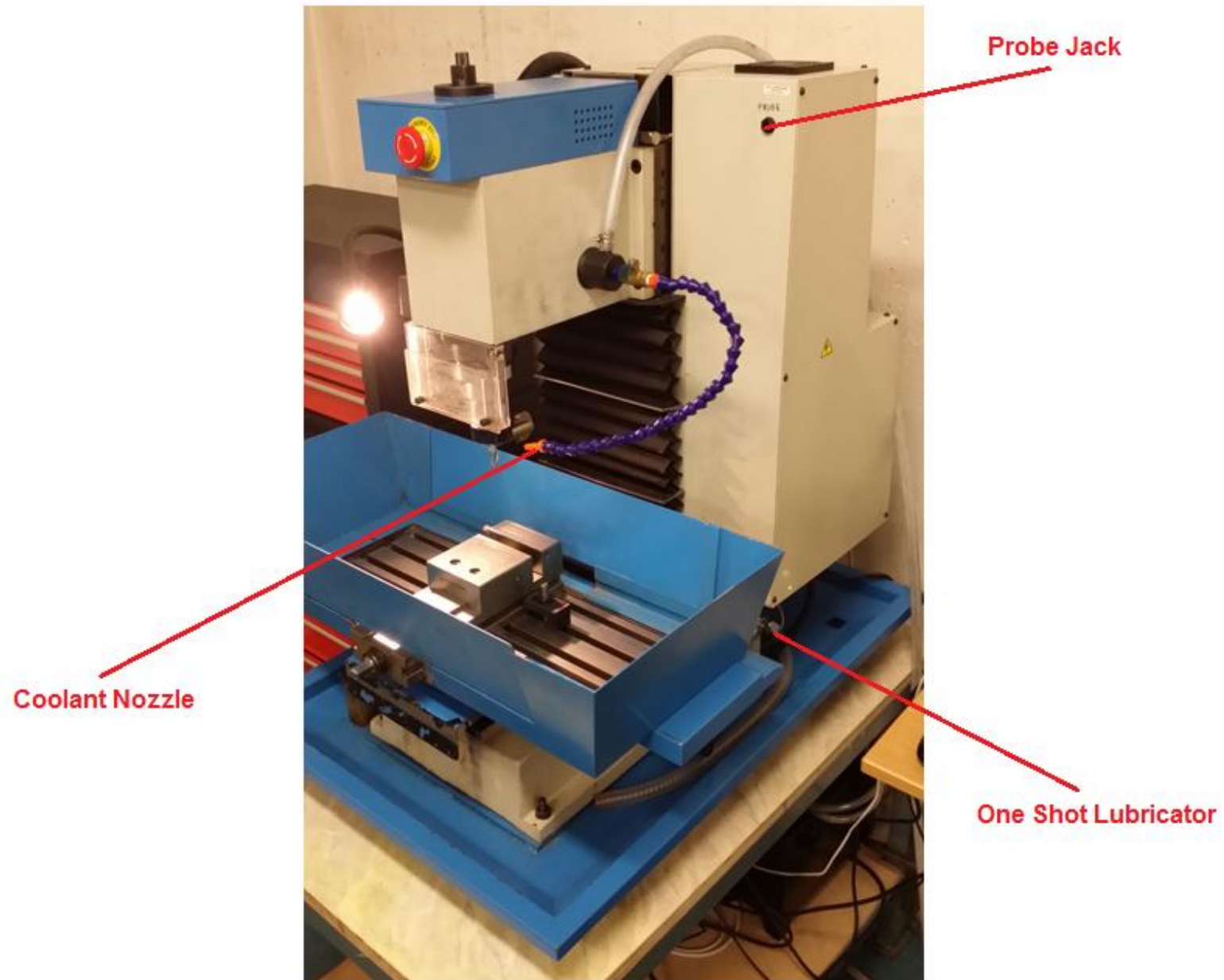
290 x 125 x 275 mm

The maximum size of stock material the CNC Mill can machine is given by its X, Y and Z travel. Keep in mind that this is further reduced by the tool diameter and the length of the tool and toolholder.

The illustration and the description on the next pages points out the basic parts of the CNC Mill that is essential to understand what they are in order to operate the machine.







On/Off Switch:	The main switch to turn on the power to the machine.
Table:	The working area that the CNC Mill can work on, this is what you clamp your work onto. Moves in the X and Y direction.
Spindle:	The spindle motor of the machine. This is where you attach the tool or toolholder that is to be used.
Drawbar:	Used to clamp the tool/toolholder into the spindle.
Control Computer:	The PC that controls the machine.
One Shot Lubricator:	The lubrication system of the machine. Pull and release the handle to lubricate the machine. This should always be done before the machine is to be used!
Safety Shield:	Protective shield against the rotating tool. Will stop the machine if opened when it is running. Must be closed in order for the machine to operate!
Emergency Stop:	Emergency button, will stop the machine if pressed. Must be reset in order for the machine to operate!
Head:	The structure that contains the spindle motor. Moves up and down, is controlled by the Z axis
Coolant Nozzle:	The nozzle where the coolant / compressed air is sprayed out from.
Coolant Tank:	The tank in which the coolant is stored.
Work Light:	Provides light at the cutting tool.
Probe Jack:	The jack in which the Auto Tool Zero Probe and the Touch Probe plugs into.
Tool Cabinet:	Tools and accessories to the CNC Mill are stored in here.

Tools and Toolholders

There are many tools available to use with the CNC Mill. Some of the tools are attached to the machine directly by mounting them in the spindle, other tools such as endmills and drills can only be attached to the machine by the use of a toolholder.

Figure 11 illustrates the available tools and toolholders that fit in the R8 spindle of the machine, Figure 12 illustrates tools mounted in toolholders and Figure 13 illustrates the assembly of a collet chuck.



Fig 11: R8 Tools and Toolholders

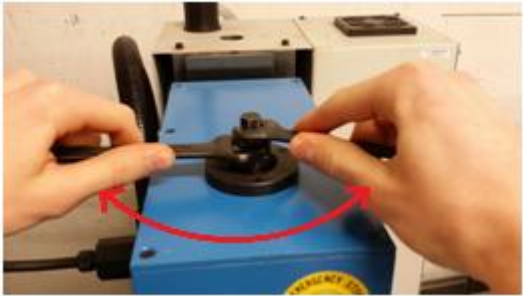


Fig 12: Tools Mounted in Toolholders



Fig 13: Assembly of Collet Chuck

A R8 tool or toolholder is mounted/unmounted into the spindle in the following way.

Attach Tool / Toolholder in R8 Spindle	
Loosen drawbar with wrenches.	
Unscrew it as far as possible by hand.	
Give the drawbar a tap in order for the tool / toolholder to release from the spindle cone.	
Hold the tool / toolholder with one hand, unscrew the drawbar with the other hand.	
Remove the tool / toolholder.	
Insert new tool / toolholder.	
Tighten the drawbar.	

A tool is mounted in a Collet Chuck in this way.

Attach Tool in Collet Chuck

Hold the collet chuck still with the hook tool and loosen the nut with the appropriate wrench.



Remove the nut completely.

Choose the correct collet for the tool.

Insert the tool and tighten the nut.



Accessories

The Auto Tool Zero probe can be used to set the Z axis zero point on the surface the probe is located on. The Probe is plugged into the jack labeled 'Probe' on the front on the machine. In Figure 14, the Z axis zero point will be set on top of the vise.

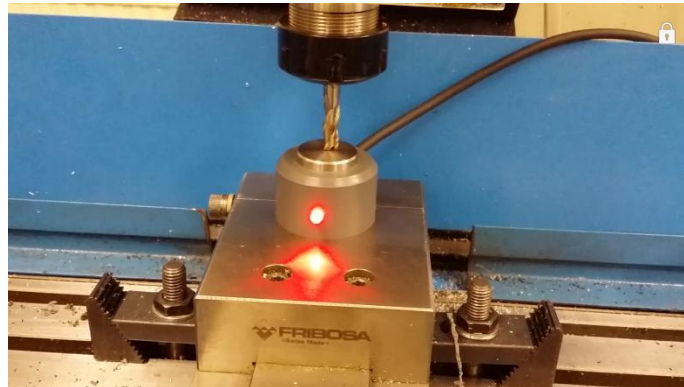


Fig 14: Auto Tool Zero Probe in use

The Touch Probe can be used to locate the edges of a part / stock. It can also be used to automatically find the center point of a hole. Figure 15 and Figure 16 illustrates the use of the probe. The Touch Probe is also plugged into the 'Probe' jack.

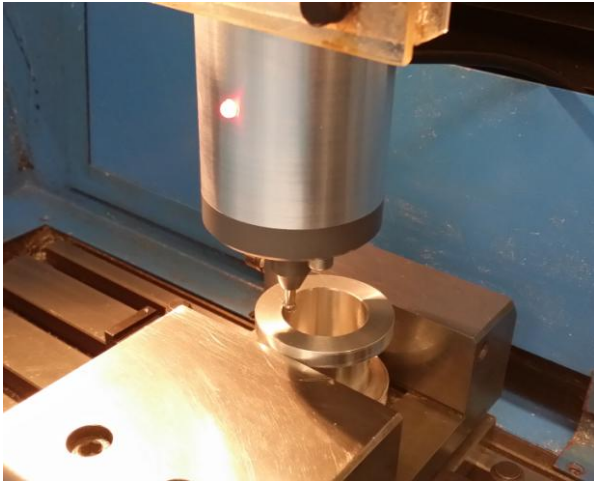


Fig 15: Touch Probe locating center of hole



Fig 16: Touch Probe locating edge

Clamps are used to clamp stock or accessories such as a vise or a lathe chuck to the table. The table has T - Slots to enable clamps to be used. A clamp is assembled from a T - Slot nut, threaded pin, clamping bar and a step block.



Fig 17: Clamps

Parallels are very precise pairs of flat steel bars in various thickness and height. The parallels are intended to be positioned at the bottom of the vise to make sure that the stock material sits parallel to the table and is clamped as high up in the vise as possible.

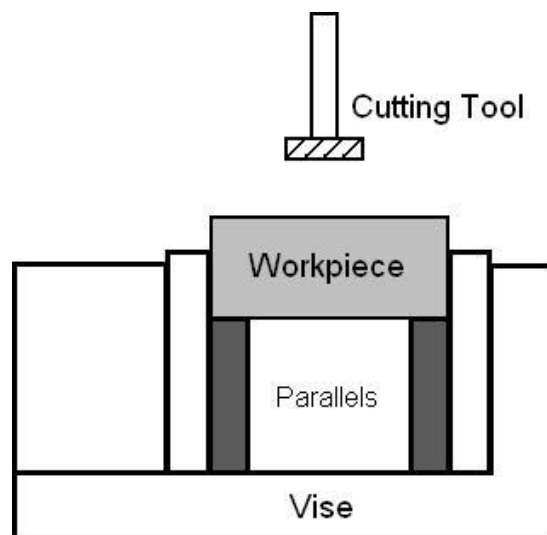


Fig 18: Parallels in Vise

A rotary axis is available to the CNC Mill. The rotary axis is plugged in to the jack labeled 'Rotary Axis' on the back side of the machine.

Mach 3 Overview

Intro

The control software of the CNC Mill is the very popular *Mach 3*. The software runs under Windows and is easy to use and has many powerful features.

To launch Mach 3:

Doubleclick the *Mach 3 Loader* shortcut on the desktop → Chose the *Student CNC* profile → Hit OK

This is the Mach 3 application.

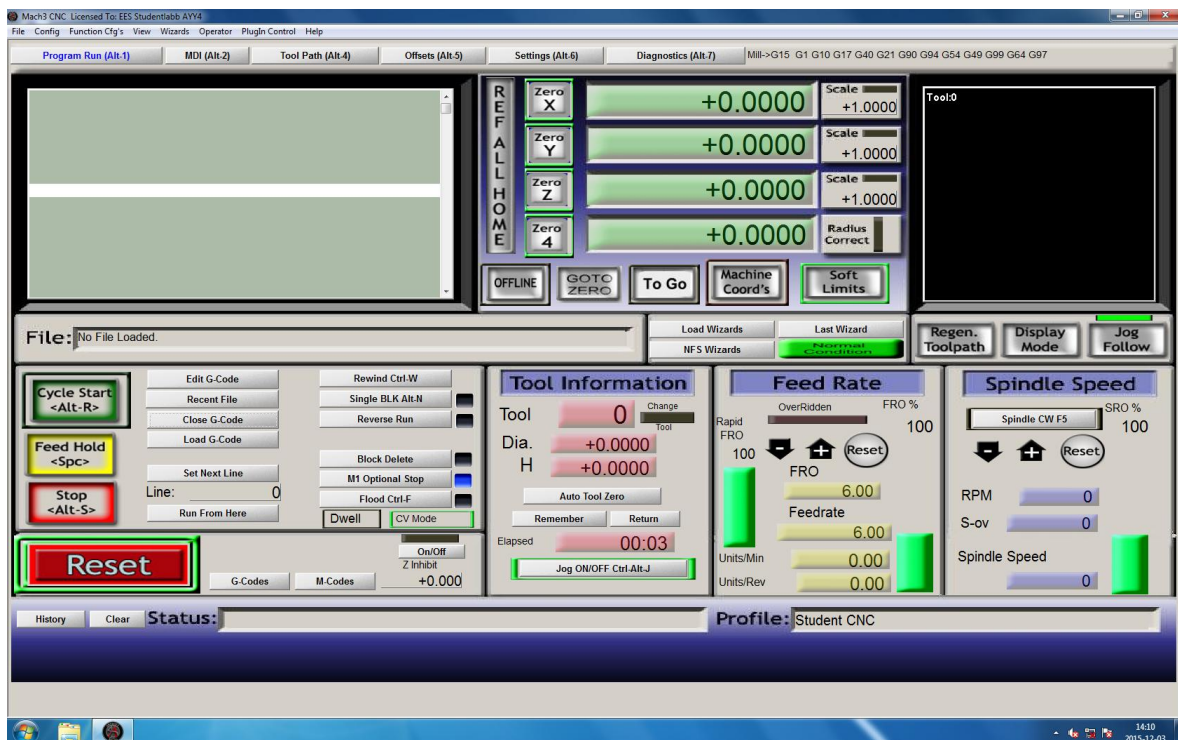


Fig 19: Mach 3 Application

Tip!

Data input in Mach 3 should be followed by ENTER, otherwise the input will not take place. Example of an input is changing the spindle speed.

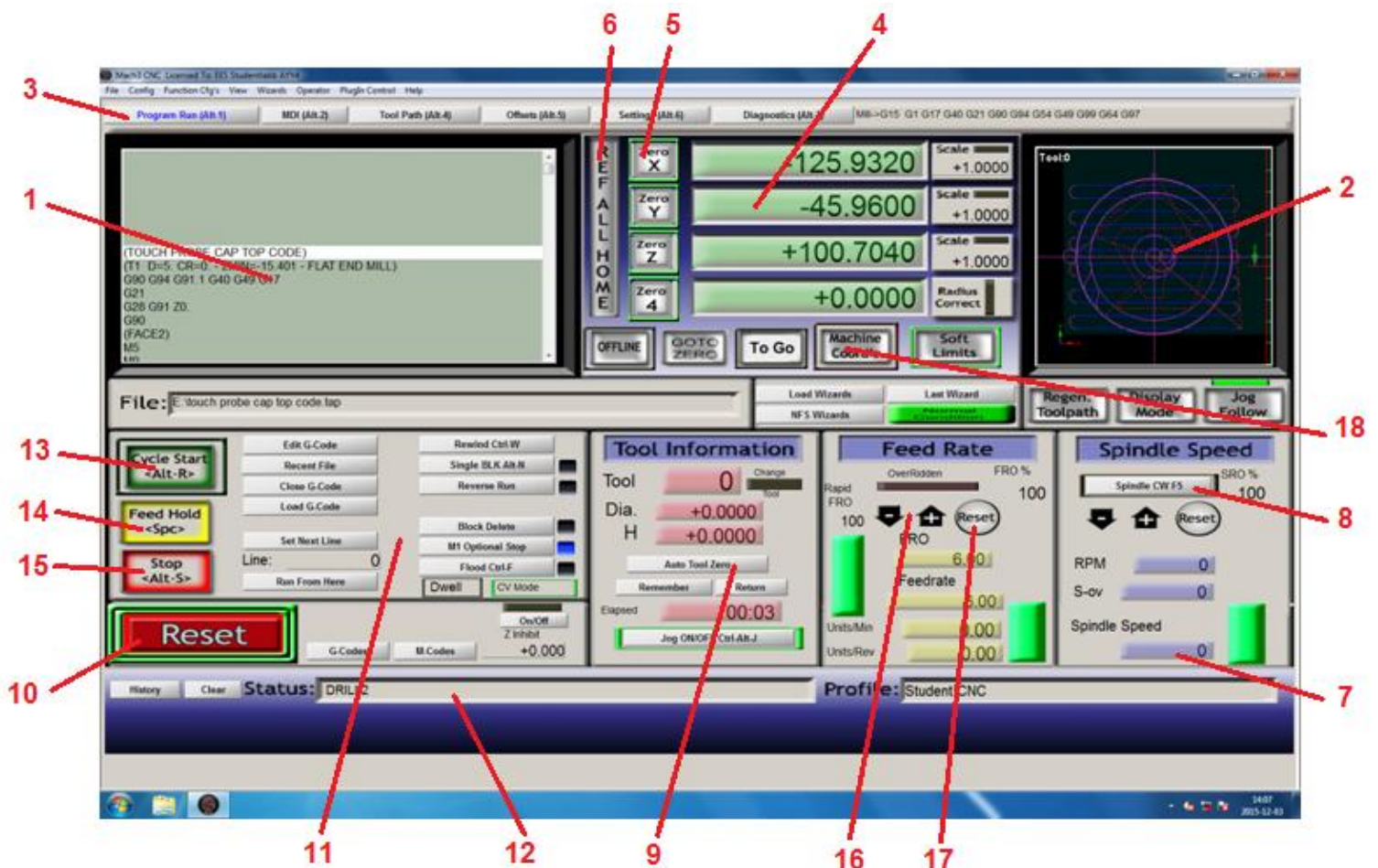


Fig 20: Mach 3 Program Run Tab

1.	G -Code window	10.	Resets Mach 3
2.	Toolpath display	11.	G - Code file operations
3.	Select tabs	12.	Status bar, displays messages
4.	Axis DRO, shows each axis position	13.	Start G - Code execution
5.	Zero buttons, zero an axis	14.	Pause a running program
6.	Performs a Homing Cycle on the machine	15.	Stop a running program
7.	Set Spindle speed	16.	Override feedrate
8.	Start/Stop Spindle	17.	Reset to feedrate specified in G - Code
9.	Auto Tool Zero Routine	18.	Change between viewing Machine Coordinates and Work Coordinates

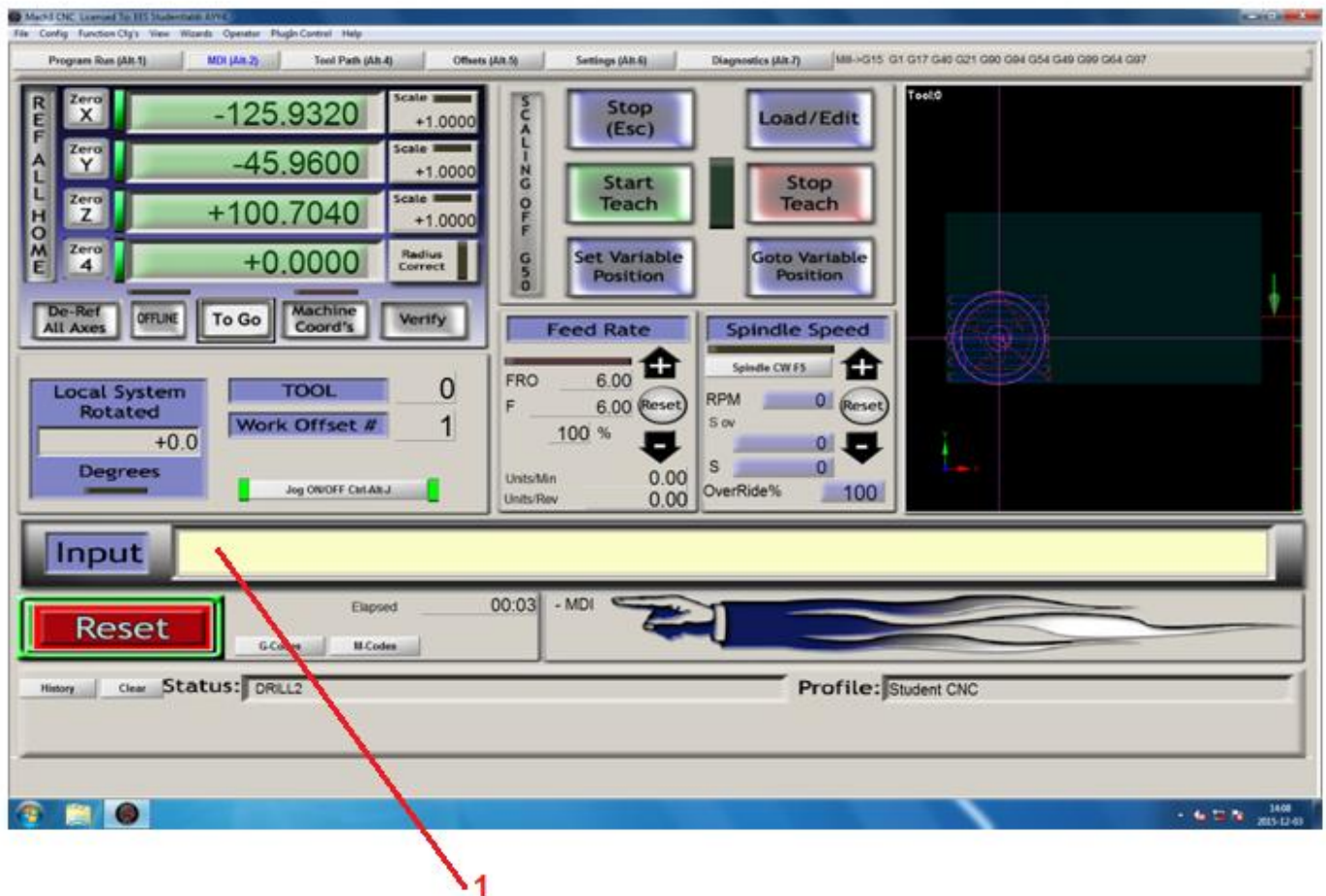


Fig 21: Mach 3 MDI Tab

1. MDI Input. Execute single line G - Code, for example 'G0 X25'.

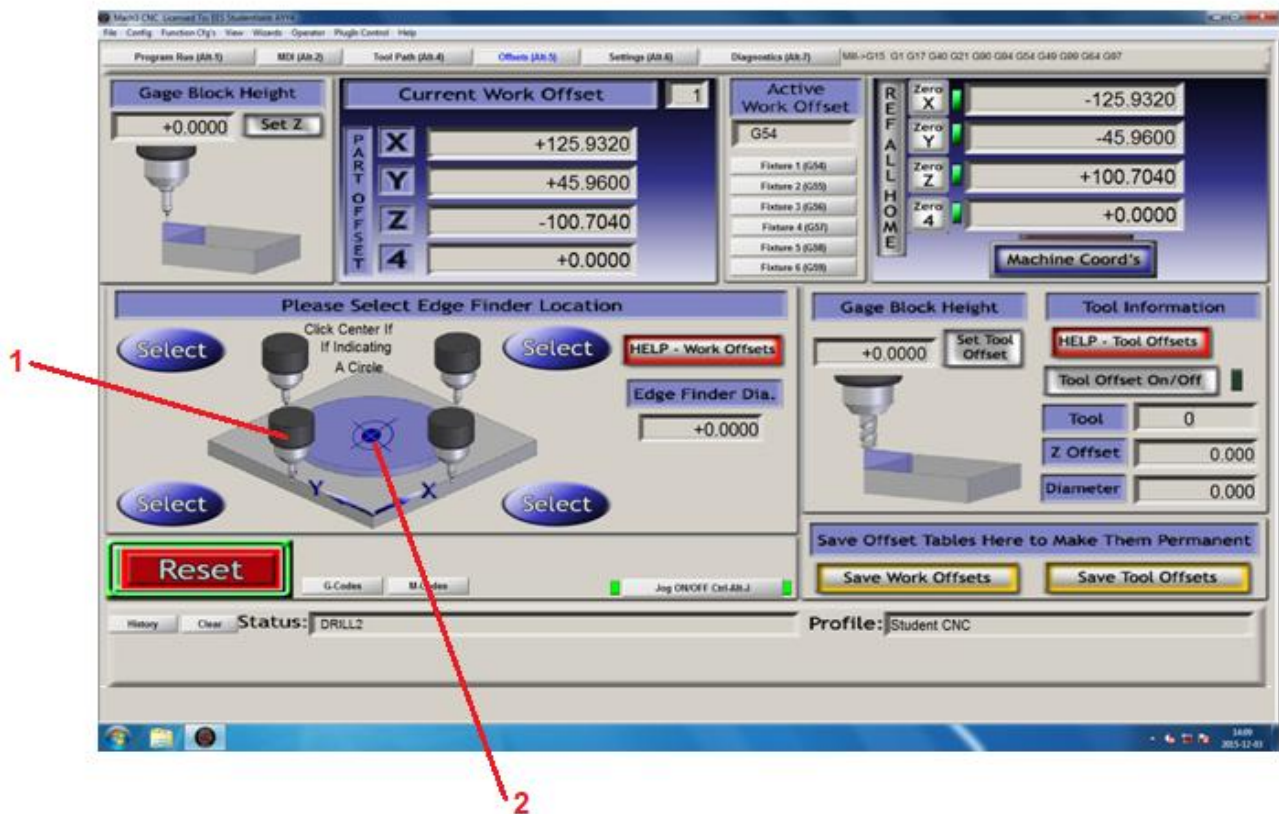


Fig 22: Mach 3 Offset Tab

1.	Find the center of a hole with the Touch Probe.
2.	Find an edge with The Touch Probe. Select one of the four directions to move the probe in. The referenced one will probe in the X+ direction.

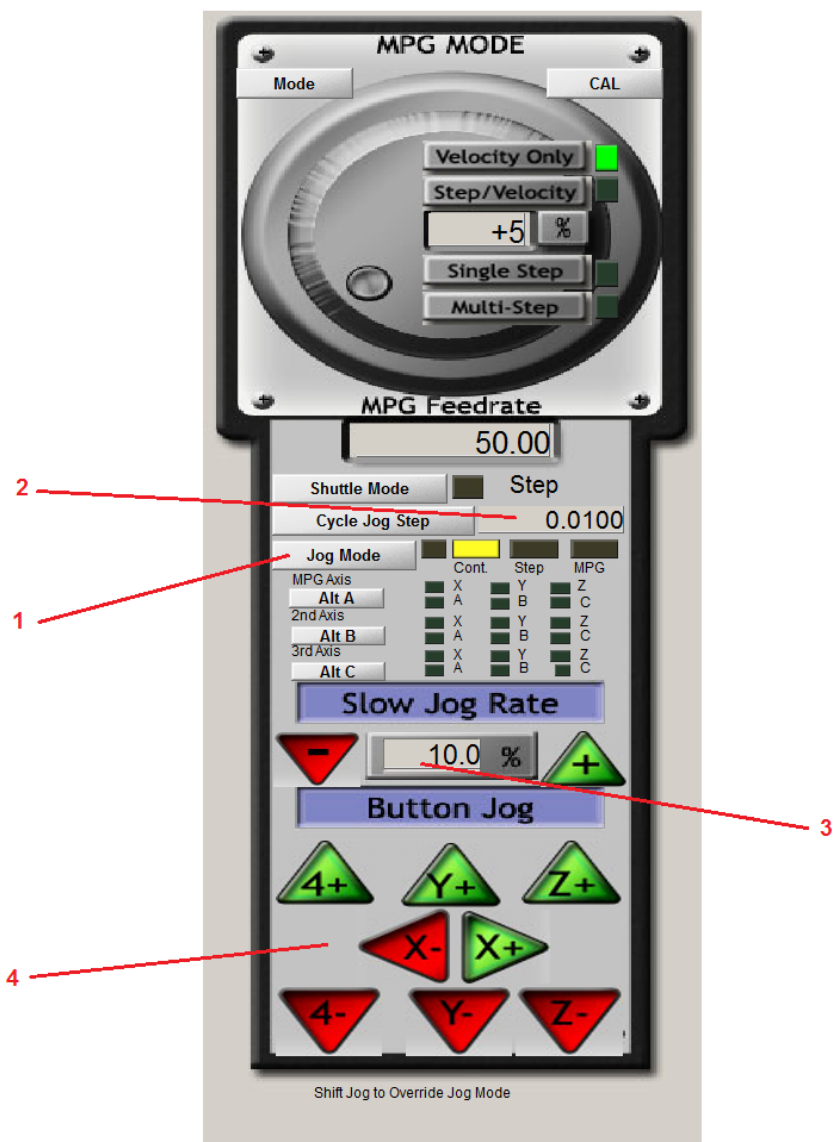


Fig 23: Mach 3 Jog Tab

1.	Change between continuous jogging and jogging in steps.
2.	Step jogging step size.
3.	Slow jogging speed, in percent of maximum jogging speed.
4.	Jog by buttons.

Jogging

The mill axes can be moved by the operator using the keyboard keys, this is called jogging the machine. Full speed jogging is accomplished by holding the SHIFT key. The Jog screen with jogging settings is shown / hidden by pressing the TAB key.

X+	Arrow Right
X-	Arrow Left
Y+	Arrow Up
Y-	Arrow Down
Z+	Page Up
Z-	Page Down

Setup Work Coordinate System Origin

The Work Coordinate system is setup by moving the endmill to the place of where the operator would like to place the origin of the Work Coordinate System, and then Zero all the coordinates values of the three axis X, Y and Z.

The accuracy required of the position of the origin of this coordinate system can vary.

If the stock material is sufficiently oversized of the part to be milled, the position of the origin can be eyeballed. The endmill is simply positioned approximately where the origin is set in the CAD/CAM program and then the X, Y and Z axis is zeroed.

If more precision is required, the Auto Tool Zero probe can be used to find the Z height of a surface and the Touch probe can be used to find edges and centers of holes. In this way the origin of the Work Coordinate System can be set exactly at the tip of a corner for an example.

MDI Overview

The Manual Digital Input (MDI) function allows for input and execution of one G -Code instruction at the time. This is useful to simple operations like moving the mill away to access the finished part, start or stop the spindle or making a simple series of cuts.



The MDI box is found on the MDI tab.

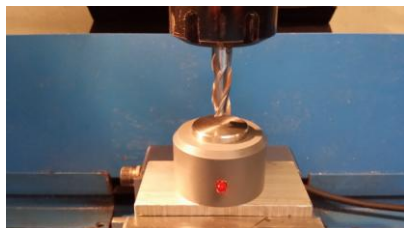
Wizards Overview

Wizards are simple programs built into Mach 3 that can generate G - Code for simple operations like mill a hole or a pocket. There are quite a few included and they cover many operations.

The Wizards can be accessed from the 'Wizards' drop down menu.

Using the Auto Tool Zero Probe

The Auto Tool Zero probe finds the Z position of the surface it is placed on and sets the Z axis zero point there.

Using the Auto Tool Zero Probe	
Plug the probe into the probe jack.	
Test that it is working by touching the endmill with the probe. The red LED on the probe should light up on contact. If not, something is not working and the probe should not be used.	
Position the probe on the surface and the endmill 10 mm above the probe.	

Click the Auto Tool Zero button.	 The image shows a button labeled 'Auto Tool Zero' at the top. Below it is a photograph of a CNC machine's tool head positioned over a metal workpiece. A red laser dot is visible on the workpiece, indicating the tool's position.
The Z axis zero point will now be set on the surface and the tool is positioned 50 mm above.	

Using the Touch Probe

The Touch Probe can be used to find edges and centers of holes.

Finding an Edge	
Plug the Touch Probe into the probe jack. Make sure that the probe is working by pressing the probe tip sideways, the red LED should light up. If not, the probe is faulty and cannot be used.	
Position the probe a bit away from the edge and adjust the Z height so that the tip of the probe is in height with the edge.	 The image shows a CNC machine's tool head with a touch probe. The probe is positioned over a metal workpiece that has a smiley face shape machined into it. The probe tip is just above the surface of the workpiece.
Select the appropriate button for probing in the direction of the edge and click it.	 The image shows a close-up of the touch probe tip, which is a small, cylindrical metal probe with a sharp point.

The probe tip will now be positioned directly above the edge.

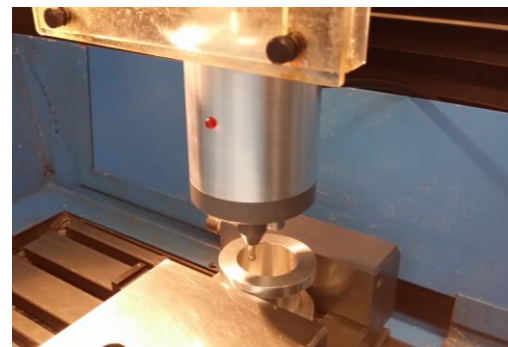


Finding the Center of a Hole

Plug the Touch Probe into the probe jack.

Make sure that the probe is working by pressing the probe tip sideways, the red LED should light up. If not, the probe is faulty and cannot be used.

Position the probe in the hole and adjust the Z height so that the tip of the probe is in height with the edges.



Click the center button.



The spindle will now be centered in the hole.



CAD/CAM Software Overview

Intro

The design of a part start by drawing it in the CAD program. The part should be drawn as a 3D model. When this is done the next step is to use the CAM part of the software to specify the cutting operations that should be done in order to produce the part from a stock material, this can then be exported as a G - Code file to be executed on the CNC Mill.

The workflow is thus as follows:

3D Modeling → Define Stock Size → Specify Cutting Operations → Export G - Code

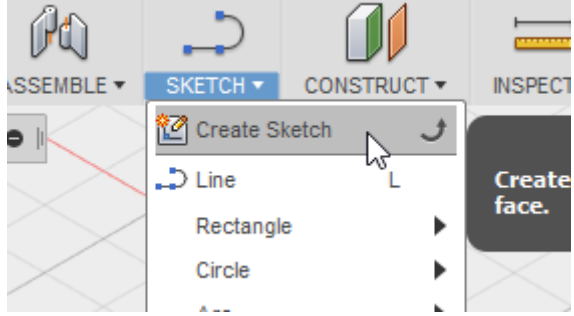
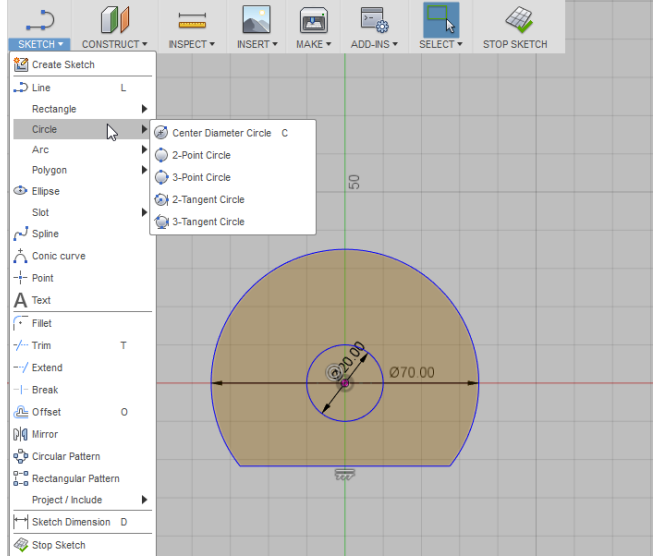
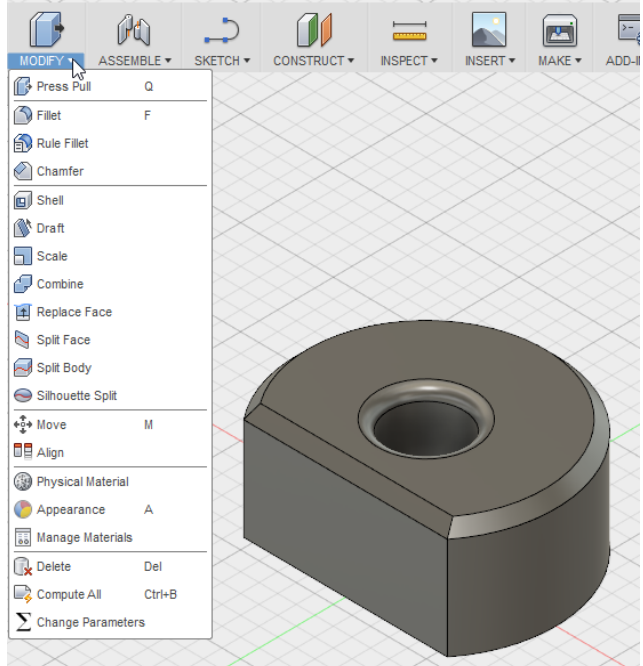
Autodesk Fusion 360

Autodesk Fusion 360 is a CAD and CAM software package that can be used to do 3D modeling and generate toolpaths / G-code to be executed on the CNC mill.

The software is free for students and can be downloaded from Autodesk after creating an Autodesk account.

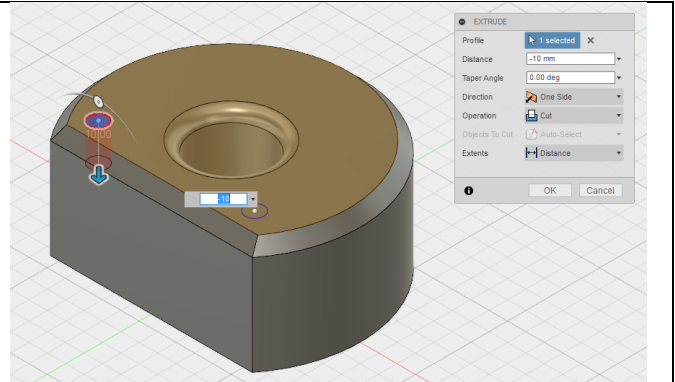
Autodesk Fusion 360 Design Workflow

CAD Modeling in Autodesk Fusion 360

1.	Start Autodesk Fusion 360, a new file 'Untitled' will be created automatically. Save it in a project folder and give it a describing name. Create a new sketch and select the plane to draw it in.	
2.	Draw the 2D geometry of the part. Use the drawing tools in the 'SKETCH' menu. When the sketch is done, return to 3D view by clicking 'STOP SKETCH'.	
3.	Use the tools in the 'MODIFY' menu to pull the sketch into a 3D geometry and modify it. Press Pull = Raise 2D into 3D Chamfer = Create a chamfered edge Fillet = Create an edge with a radius	

4.

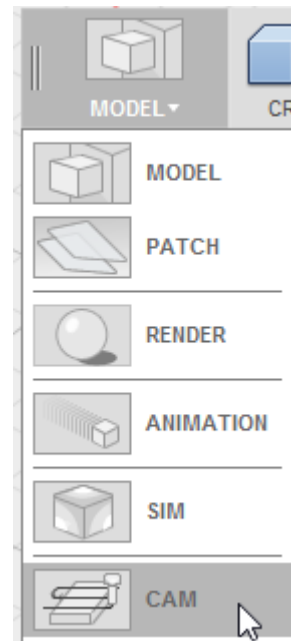
Create additional sketches if necessary. Modify them into 3D geometry.



CAM Process in Autodesk Fusion 360

1.

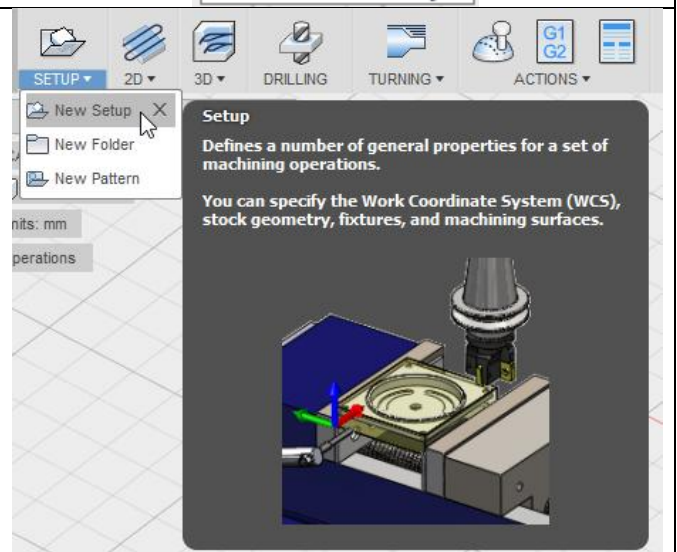
Select CAM mode.

**2.**

Create a new setup.

Select the origin point under the 'Setup' tab.

Specify the stock material shape and size under the 'Stock' tab.



3.

Create the necessary cutting operations.

Choose the cutting operation from the 2D menu.

Select the cutting tool to be used.

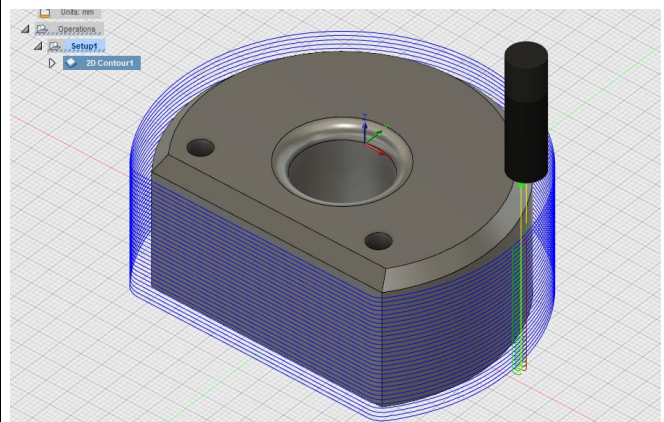
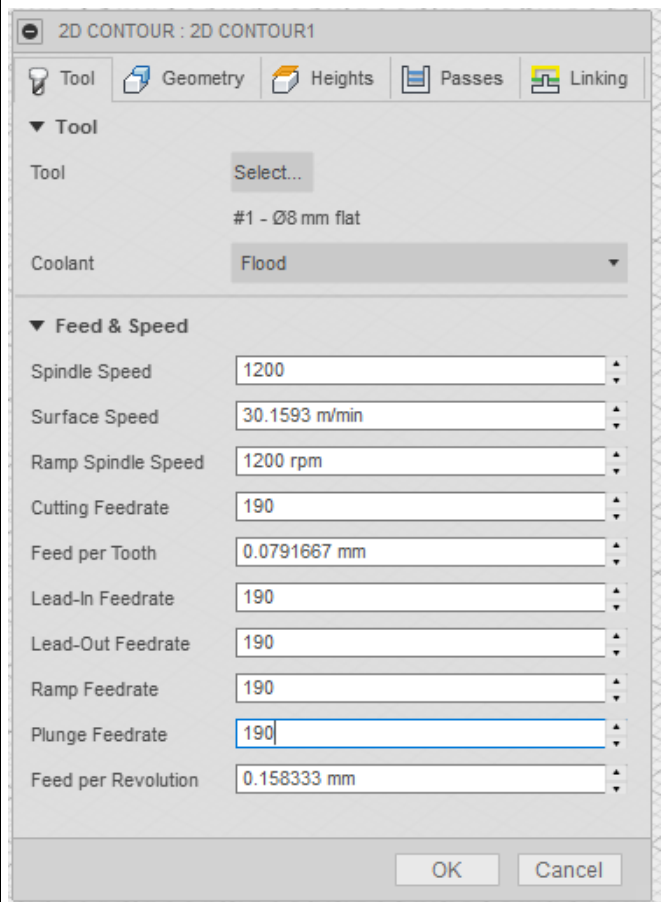
Fill in the Spindle Speed, Ramp Spindle Speed, Cutting Feedrate, Lead-In Feedrate, Lead-Out Feedrate, Ramp Feedrate and Plunge Feedrate according to the Speeds and Feeds Table.

Select the geometry to be cut.

Select the heights of the depth of the cut.

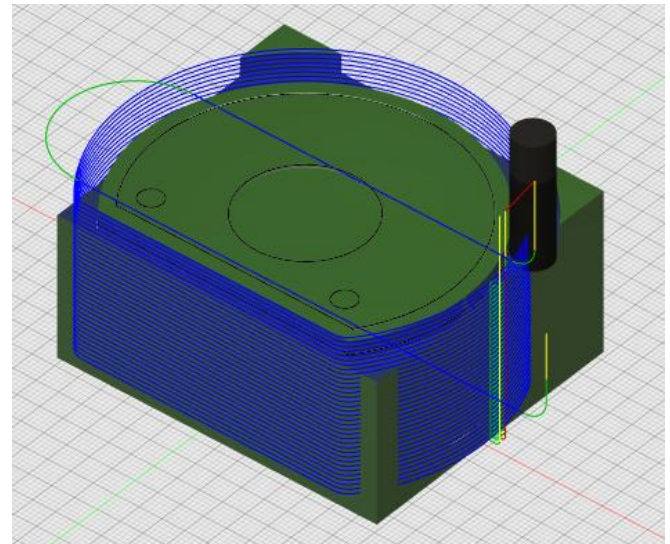
Select the depth per pass under 'Passes' tab.

Keep in mind how the part should be fixtured / clamped to the table. It is common to machine it halfway and then flip it around in order to machine the second half.



4.

Simulate to make sure that the toolpaths are correct.

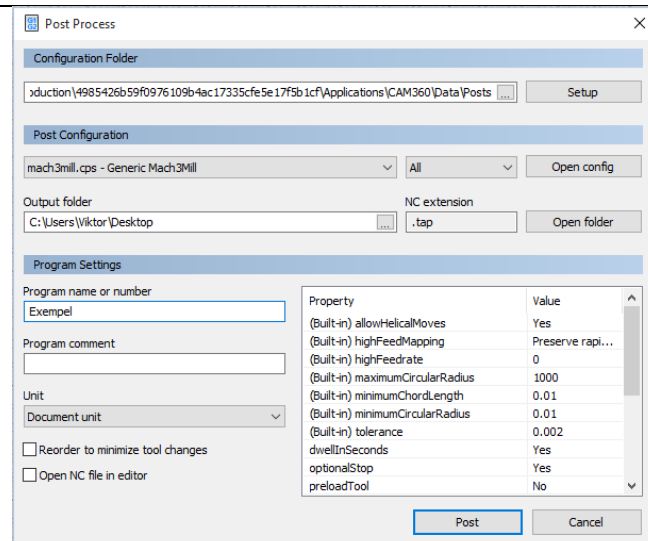
**5.**

Generate the G -Code by selecting 'Post Process'

Select 'mach3mill.cps' under 'Post Configuration'

Give the program a name under 'Program name or number'

Hit OK , save the file and head to the CNC Mill!

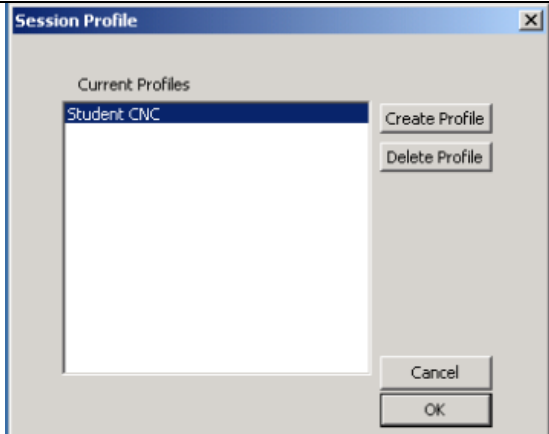


Workflow of Machining a Part

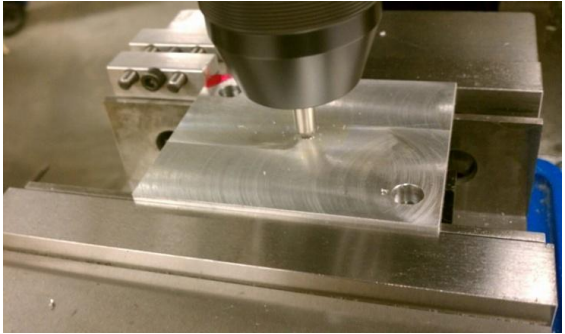
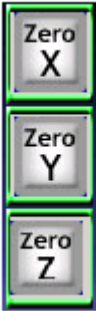
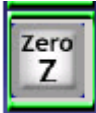
This section describes step by step how to run a typical job on the CNC machine.

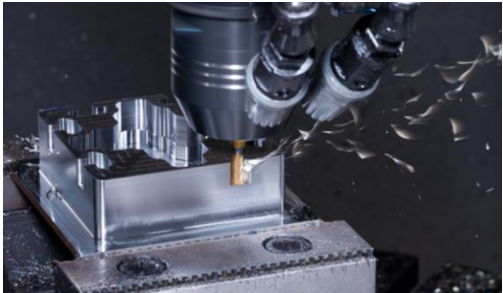
Make sure to take the time to read the previous sections, you will not be successful in operating the machine if you don't understand the basics of CNC machining. It is also important to follow the safety rules in order to not harm yourself or others.

Read and understand the Safety Rules before operating the machine!

Workflow of Machining a Part		
1.	Generate the G - Code either by: - Autodesk Fusion 360 - Mach 3 Wizards - Writing it manually Make sure that the Feeds, Speeds and Maximum Depth Of Cut are set accordingly to the 'Speeds and Feeds' table.	
2.	Find a suitable stock material. Cut it to be a bit oversized to the part that you are machining.	
3.	Turn on the computer	
4.	Turn on the power to the mill	
5.	Pull and release the oiler to lubricate the machine	
6.	Start Mach 3 by opening <i>Mach 3 Loader</i> → Chose the <i>Student CNC</i> profile → Hit OK	

7.	Reset Mach 3 by clicking the 'Reset' button	
8.	Reference the machine by performing the homing cycle. Press the 'REF ALL HOME' button. This will reference the Machine Coordinates, making sure that the offsets works.	
9.	Mount the stock material to the table either by: Clamping it directly to the table with a safety sheet of material underneath. Clamping it in the vise. Use parallels to make sure it is parallel to the table. Clamping it in the lathe chuck.	
10.	Insert the cutting tool into the spindle. Make sure that it is securely tightened	

11.	Set the origin point of the coordinate system (G54 Work Offset). Do this by jogging the machine until the tip of the endmill is in the appropriate spot in the XY plane and directly above the stock material. Zero the X, Y and Z axis. The Auto Tool Zero Probe can be used to automatically and very accurately set the Z height.	 
12.	Move the Z axis up 50 mm. Do this with the G - Code 'G0 Z50' in the MDI box on the tab MDI". Then zero the Z axis. This is so that the program first can be tested without cutting into the material. This is known as 'cutting air'. This is to determine that the program works as indented before the real cut is made.	 
13.	Load the G - Code into Mach 3 by clicking 'Load G - Code'.	
14.	Start the program by pressing 'Cycle Start'.	
15.	When the program is verified to work correctly: Stop the program with the 'Stop' button. Rewind the program with the 'Rewind' button.	

	Lower the Z axis 50 mm with the G - Code command 'G0 Z-50'. Zero the Z axis. Start the program by pressing 'Cycle Start'.	
16.	Monitor the machine while it is working. Do not leave it unattended. You need to be around and stop it if something goes wrong. When it is running, chips need to be blown away and cutting oil applied if steel materials are cut.	
17.	When you are done using the machine it should be closed down in this order: Close Mach 3. If prompted with 'Fixture Save?' choose Yes. Turn off power to the machine. Turn off the computer.	
18.	Cleaning. The machine should be completely cleaned when you are done using it. Use a dustpan and brush to clean the chips. Wipe off oil from the machine with paper towels. Vacuumclean the machine and the floor.	
19.	Clean all the tools you have used and put them back in their designated place in the tool cabinet.	

Speeds and Feeds Table

Material	Tool Diameter (mm)	Feed Rate (mm/min)			Plunge Feed Rate	Spindle RPM	Maximum Cutting Depth (mm)
		Cutting Flutes					
		2	3	4			
Mild Steel	3	95	140	190	50	3100	0.3
	4	95	140	190	50	2400	0.4
	6	95	140	190	50	1600	0.5
	8	95	140	190	50	1200	0.8
	10	95	140	190	50	950	1
	12	95	140	190	50	800	1
	15	95	140	190	50	700	1
	20	95	140	190	50	750	0.5
	63	95	140	190	50	1000	0.5
Hard Steel	3	40	60	75	25	1300	0.15
	4	40	60	75	25	1000	0.2
	6	40	60	75	25	600	0.25
	8	40	60	75	25	500	0.35
	10	40	60	75	25	400	0.5
	12	40	60	75	25	300	0.5
	15	40	60	75	25	250	0.75
	20	40	60	75	25	500	0.5
	63	40	60	75	25	1000	0.25
Aluminum	3	238	360	500	100	5000	0.5
	4	238	360	500	100	5000	0.5
	6	238	360	500	100	4000	0.8
	8	238	360	500	100	3000	1
	10	238	360	500	100	2400	1
	12	238	360	500	100	2000	1
	15	238	360	500	100	1600	1
	20	238	360	500	100	1200	1
	63	238	360	500	100	1000	1

Material	Tool Diameter (mm)	Feed Rate (mm/min)			Plunge Feed Rate	Spindle RPM	Maximum Cutting Depth (mm)
		Cutting Flutes					
		2	3	4			
Plastic	3	95	140	190	100	3100	0.3
	4	95	140	190	100	2400	0.4
	6	95	140	190	100	1600	0.5
	8	95	140	190	100	1200	0.8
	10	95	140	190	100	950	1
	12	95	140	190	100	800	1
	15	95	140	190	100	700	1
	20	95	140	190	100	500	1
	63	95	140	190	100	1000	1

Prefix Reference

A	A axis of machine
B	B axis of machine
C	C axis of machine
D	Tool radius compensation number
F	Feedrate
G	G - Code
H	Tool length offset index
I	X axis offset for arcs
	X offset in G87 canned cycle
J	Y axis offset for arcs
	Y offset in G87 canned cycle
K	Z axis offset for arcs
	Z offset in G87 canned cycle
L	Number of repetitions in canned cycles/subroutines
	L1 / L2 tool offsets settings / fixture offset (with G10)
M	M -Code
N	Line number
O	Subroutine label number
P	Dwell time in canned cycles
	Dwell time with G04
	Tool / Fixture number (with G10)
	Tool radius (G41, G42)
Q	Feed increment in G83 canned cycle
	Repetitions of subroutine call
R	Arc radius
	Canned cycle retract level
S	Spindle speed
T	Tool selection
X	X axis of machine
Y	Y axis of machine
Z	Z axis of machine

G - Code Reference

G0 Rapid positioning
G1 Linear interpolation
G2 Clockwise circular/helical interpolation
G3 Counterclockwise circular/Helical interpolation
G4 Dwell
G10 Coordinate system origin setting
G12 Clockwise circular pocket
G13 Counterclockwise circular pocket
G15/G16 Polar Coordinate moves in G0 and G1
G17 XY Plane select
G18 XZ plane select
G19 YZ plane select
G20/G21 Inch/Millimeter unit
G28 Return home
G28.1 Reference axes
G30 Return home
G31 Straight probe
G40 Cancel cutter radius compensation
G41/G42 Start cutter radius compensation left/right
G43 Apply tool length offset (plus)
G49 Cancel tool length offset
G50 Reset all scale factors to 1.0
G51 Set axis data input scale factors
G52 Temporary coordinate system offsets
G53 Move in absolute machine coordinate system
G54 Use fixture offset 1
G55 Use fixture offset 2
G56 Use fixture offset 3
G57 Use fixture offset 4
G58 Use fixture offset 5
G59 Use fixture offset 6 / use general fixture number
G61/G64 Exact stop/Constant Velocity mode
G99 Rapid Height By R HeightG68/G69 Rotate program coordinate system
G70/G71 Inch/Millimeter unit
G73 Canned cycle - peck drilling
G80 Cancel motion mode
G81 Canned cycle - drilling
G82 Canned cycle - drilling with dwell
G83 Canned cycle - peck drilling
G85/G86/G88/G89 Canned cycle - boring
G90 Absolute distance mode
G90.1 Absolute IJK mode
G91 Incremental distance mode
G91.1 Incremental IJK mode
G92 Offset coordinates and set parameters
G92.x Cancel G92 etc.
G93 Inverse time feed mode
G94 Units Per Min.
G98 Rapid Height By Z Height

M - Code Reference

M0	Program stop
M1	Optional program stop
M2	Program end
M3	Start spindle clockwise
M4	Start spindle counterclockwise
M5	Stop spindle
M6	Tool change
M7	Mist coolant on
M8	Flood coolant on
M9	All coolant off
M30	Program end and rewind
M47	Repeat program from first line
M48	Enable speed and feed override
M49	Disable speed and feed override
M98	Call subroutine
M99	Return from subroutine/repast